Computational Fluids & Structures Engineering

Reference	GSTP-ES
Revision	1
Date	03. 07 2020
Reg. No.	
Page	1 of 13

Report

Executive Summary: GSTP Study on the Modelling of hot plumes with particles for improved launcher applications

J.B. Vos and D. Charbonnier

CFS Engineering
EPFL Innovation Park, Batiment-A,
CH-1015 Lausanne

Internal copies

External copies

PREPARED BY: Date Visa Name J.B. Vos	REVIEWED BY: Date Visa Name	APPROVED BY: Date Visa Name

Change Log

ISSUE	PAGE	CHAPTER	DESCRIPTION	DATE	VIS
1.0	all	all	First Issue	03.07.2020	J.B. Vos

Contents

1 Introduction

Europe's access to space today relies primarily on the Ariane 5 family of heavy lift launchers and the Vega launcher for small payloads. The successful development of these launchers depends on finding solutions in the critical and challenging areas of propulsion and aero-thermodynamics, which are the key elements of any launch vehicle. In the past aerodynamic and aero-thermodynamic development work was almost exclusively based on the use of engineering and empirical methods. Today the use of Computational Fluid Dynamics (CFD) has matured to the point where it can provide valuable physics based input where in the past empirical or engineering methods required to strongly simplifying the physics involved. In addition, experimental testing is extremely expensive, time consuming and often it is impossible to simulate the real flight conditions. As a result the data obtained from these experiments is only partially useful and approximate methods need to be used to extrapolate these data to real flight conditions.

For these reasons CFD based methods are considered as the best solution to accelerate and refine the design process of launch vehicles. However, CFD methods have their own shortcomings and limitations. First, CFD simulations are time consuming, in particular for unsteady flow simulations. Second CFD solvers employ physical models, as for example for turbulence and combustion. Although much progress has been made in turbulence modelling for unsteady flows over the last 15 years, uncertainties in computed results remain important. Combustion modelling, and in particular the influence of turbulence on the combustion process, is an area where still much research is needed. When using a solid propellant additional complexities arise due to the presence of particles in the exhaust gases, and the importance of radiation phenomena that influence the combustion process.

The work carried out in the Hot-Plume project is concerned with the after body flow of launch vehicles (also called the base flow region, see Fig. 1.1. This flow is characterized by large regions of unsteady separated flow induced by the abrupt changes in geometry of the vehicle. In this region hot gases from the nozzle exit mix with the cold flow coming around the launch vehicle leading to very complex aerodynamic phenomena which today are only poorly understood. As a result there exists a large area of uncertainty in launch vehicle design.

Over the past 15 years a large number of experimental and numerical studies were performed to better understand the complex flow phenomena in the base region. However, wind tunnel tests in Europe use cold plumes for the flow from the nozzle exit (jet-on conditions), but in-flight measurements on the Ariane5 and on the inaugural flight of VEGA have shown that this approach underestimates the base pressure and hence over estimates the vehicle drag. Furthermore, measurements during the first flight of VEGA showed that the predictions performed in the design phase largely over-predict heat loads observed in flight. This implies an influence of the plume temperature on the complex flow field surrounding the base region, which in turn implies an uncertainty in the heat loads and fluctuating loads on the nozzle itself.

Experimental results using a jet flow of $800\ K$ showed an increase of 15% in base pressure compared to the results using a jet flow of $273\ K$ [1]. Similar trends were found during the first flights of the Space Shuttle. Base pressure tests conducted prior to launch at NASA Lewis and Calspan used a cold plume to save costs and to expedite the design process. The overestimate in drag observed during initial flights triggered a post-flight hot plume test campaign in the 16T tunnel at AEDC (unpublished data). The most important external factors that affect the pressure in the base flow region of a launch vehicle are [2]:

Figure 1.1: Ariane 5 ECA V189 Lift Off, figure courtesy of ESA/CNES/Arianespace, S. Martin 2009

- Nozzle pressure ratio and nozzle length
- Free-stream parameters including Mach number and Reynolds Number
- Effect of separation induced by compression waves
- Flow topology in the base region
- State and structure of mixing layers of the main flow and the jet
- Action of additional surfaces; control surfaces, protruding elements
- Fluid properties (temperature, pressure, chemical composition) of the jet flow coming out of the nozzle

CFD solvers have now become standard tools in the design of launch vehicles. However their application to base flow analysis is mostly for so called cold plumes. These CFD studies are in good agreement with experimental data, and are able to simulate the nozzle deformation due to the unsteady pressures fluctuations in the base region [3]. CFD simulations of the base flow region for a hot plume are almost non-existent for several reasons. Modelling the hot gases of the jet flow coming out of the nozzle is a challenging task, and requires detailed information on the combustion process and the chemistry. The state of the chemistry varies from frozen to full non-equilibrium chemistry, depending on location in the flow. Often the flow coming out of the nozzle is taken as frozen, but there are indications that this is not always a valid assumption, in particular when solid particles are present in the flow. Taking into account detailed chemistry of the species involved in the combustion process increases the costs of the CFD simulation by at least an order of magnitude. And finally hardly any experimental data is available for validation of the physical models used.

For launchers using a solid propellant (like VEGA) an additional complexity exists. Solid propellants often include Aluminium particles to enhance the combustion process, and the flow of these particles needs to be modelled in the CFD simulation. In addition, radiative heat transfer from these particles is important and needs to be taken in to account since it affects the combustion process. This kind of nozzle flow has not been investigated experimentally in depth; at least there is hardly any information available in the open literature. ESA has an on-going TRP activity on this subject (Experimental Modelling of Alumina Particulate in Solid Booster) that might provide valuable experimental data for validation.

An even more challenging task is the modelling of the interaction of the hot retro rockets nozzle flows with the opposing external flow at the separation of different launcher stages. This interaction is inherently unsteady, and slight asymmetries in the interaction flow field may lead to perturbations that require corrections. Valuable experimental data is available (for cold retro rocket flows) which can be of use for validation of multiple plume interaction problems. Future ESA launcher development programs will likely include single plume stage (similar to VEGA) and multiple plume stage (similar to Ariane IV and Ariane V first stages) configurations. Data from single plume configurations cannot be directly extrapolated to the multiple plume configurations and so any wind-tunnel used for future launcher programs or useful computational method must include the possibility to consider multiple plumes simultaneously. However, the capability to correctly describe a single jet - external flow interaction is a pre-requisite for an extension to multiple plumes.

Finally the ESA Clean Space Initiative [4] demands studies of the impact of rocket launches on the atmosphere. This requires the knowledge of the composition of the exhaust gases of solid rocket motors and the detailed modelling of hot plumes for a possible integration in climate simulations.

In the GSTP Hot Plume Project the NSMB CFD solver was extended with chemistry models, a particle tracking module and with a radiation model. The extended NSMB CFD solver was used to compute the plume of the VEGA launcher [5] and to perform CFD simulations of the hot plume experiments that were made at the Instuture of Aerodynamics and Flow Technology of the German Aerospace Center DLR in Köln [6].

2 Objectives

The main objective of the GSTP project on the modelling of hot plumes with particles was to obtain a better understanding of the after body flow of launch vehicles with hot gases coming out of the nozzle exit in order to reduce design margins for launch vehicles [7].

To reach these objectives, the existing NSMB CFD code was to be extended with the following models:

- chemistry models (frozen, equilibrium and non-equilibrium) for the simulation of hot plumes
- a Lagrangian particle tracking model to track solid particles coming out the nozzle exit from VEGA type of launch vehicles
- a simple radiation model for solid particles

The activities in the GSTP project were to be carried out in the following four technical work packages:

1. Pre-assessment of the NSMB CFD code
2. Extension of the functionalities of the NSMB CFD code
3. Assessment and Validation of the NSMB CFD code
4. CFD validation using specific data

3 Summary of Activities

3.1 Pre-assessment of the NSMB CFD code

CFD simulations were made for the VEGA launcher around the 4 stage VEGA launcher at different flight regimes, ranging from transonic until supersonic conditions. Results were compared with available experimental and numerical results from the literature. The assessment study included

- a mesh sensitivity study
- a study on the influence of the bow shock adaption
- a study on the influence of the space discretization scheme
- a study on the influence of the turbulence model

The study showed that there were some uncertainties concerning the experimental results from DLR at Mach=5.29. The computed C_p obtained with the DLR TAU code and NSMB are close, they differ considerably from the measured data.

The conclusion of this study was that the NSMB CFD solver was able to simulate the VEGA launcher at different flight conditions. The mesh sensitivity study showed that a grid having around 2 Millions cells for a quarter geometry is sufficiently accurate to to compute the axial force coefficient and the pressure coefficients. This is further improved by local grid refinement and bow shock adaption. The turbulence model sensitivity study showed different behaviors for the different cases considered.

3.2 Extension of the functionalities of the NSMB CFD code

The NSMB CFD code was successfully extended with the following modelling capabilities:

- A general equilibrium flow solver that permits 2 different streams of gases. This model is similar to the CEA chemical equilibrium program [8], but it uses a different approach to solve the equations. Thermodynamic data to the equilibrium flow solver follow the NASA 9 coefficients polynomials [9]. Transport coefficients (viscosity and thermal conductivity) of the different species in the mixture are computed following the method from the Chemkin Package [10]. The conserved scalar approach (already available in NSMB) was adapted to handle flows with two different streams (nozzle exhaust stream and the flow around the vehicle). It should be mentioned that during the GSTP study NSMB was in the ESA TRP on Ablation modeling coupled to the Mutation++ Library developed at VKI [11] that permits a general chemical non-equilibrium simulation capability.
- The simulation of aluminum particles transport in the hot plume was implemented using Lagrangian particle tracking. Two different coupling approaches with the gas have been implemented, the so called one-way coupling in which it is assumed that there is no interaction between the particles and the particles have no effect on the fluid, and the two-way coupling which also assumes that there is no interaction between particles, but the effect of the particles on the fluid is taken into account. Four seeding methods to inject the particles in the fluid have been

implemented. Particles can be injected in different groups, and each group may have a different diameter and a different injection velocity.

- Radiation modeling is a vast field, and solving the Radiative Transfer Equation is almost impossible due to its dependence on the 3 spatial directions, the solid angles and the wavelengths. For this reason simplified methods are used and the so called P-1 equation [12] is one of the simplest and widely used methods. The P1-equation can be coupled to the Weighted Sum of Gray Gas (WSGG) Model [13, 14] which considers the emittance of a medium to be represented by the sum of some gray gases. Both the P1-equation as well as the WSGG models were implemented in NSMB. A correction procedure was implemented to accelerate the convergence of the P1-equation. Particle radiation was added to the P1-equation.

Calculations were made demonstrating the capabilities of each of the new modeling capabilities. Chemical equilibrium calculations for the Thrust Vector Control case showed a good agreement with results obtained using the CEA code from NASA. Calculations were also made for the VEGA launcher case comparing 3 different chemistry modeling approaches. Computed thrust levels and massflow rates are comparable. The chemical non-equilibrium calculations showed a frozen flow in the nozzle exit and in the plume region.

The particle tracking was tested for a supersonic flow case. Results are compared with experimental results and numerical studies found in the literature. Overall a good agreement between the results obtained by NSMB and the experimental results was obtained.

The P1-radiation model was used to compute the Ballistic Evaluation Motor. Both a thermal perfect gas and equilibrium flow assumption were used. The P1-model was used using both a constant absorption coefficient and with the Weighted Sum of Gray Gases (WSGG) model. Only small differences in results were observed when using stand alone and the fully coupled approach. Although the P1-model is unable to predict spectral properties and infrared contours, one could observe a qualitatively agreement between the incident radiation and IR contours when using the WSGG model.

3.3 Assessment and Validation of the NSMB CFD code

The extended NSMB solver was successfully used to simulate the VEGA nozzle and VEGA launch vehicle using coupled particle tracking and radiation for two different flight conditions.

A large number of parameters was tested in the calculations for the VEGA nozzle. The following conclusions were drawn from the simulations without radiation:

- a particle time step of 10^{-5} seconds is recommended for these calculations;
- at least 500 particles should be injected;
- the update frequency for the particle tracking should be at maximum every 5 steps;
- particles with larger diameter move more outwards in the plume compared to particles with a smaller diameter. Similar observations were found in the literature [15], [16].

More recent analysis showed that this last conclusion is not correct, larger particles remain closer to the symmetry axis when leaving the nozzle exit. The literature test cases do not concern a plume but are concerned more with the combustion chamber ([15]) or a channel flow ([16]).

The calculations with particle tracking and radiation clearly showed the influence of radiation on the plume. The plume is slightly larger when using radiation, and peak temperatures in the plume (near the plume boundary) are slightly lower. Radiation also seem to smooth the flow temperatures.

The calculations with a non-zero particle emissivity showed that the temperature of the particles is reduced which was to be expected.

The calculations for the VEGA launch vehicle showed that the radiative heat flux on the launch vehicle strongly depends on the parameters used for the radiation.

Some calculations were made assuming a chemical non-equilibrium flow using the reaction scheme of Troyes [17]. These calculations were extremely costly in terms of CPU time. Temperatures in the plume are lower compared to the chemical equilibrium calculation, also leading to lower temperatures of the particles.

3.4 CFD validation using specific data

Simulations including particle tracking and radiation were made for the hot plume experiments made in the DLR Köln VMK wind tunnel. The simulations were successful, and showed large unsteady separated flow regions just downstream of the nozzle exit. These large unsteady flow regions makes detailed comparisons difficult.

The calculations with particle tracking showed that a precise knowledge on the particle mass flow rate is important. Accounting for particles reduces the flow velocity, and increases the temperature on the symmetry lines. The particle mass flow rate influences the particle velocity at the nozzle exit. The higher the particle mass flow rate the lower the fluid velocity, and this lower fluid velocity will in return also reduce the particle velocity. For one calculation measured particle velocities near the symmetry line are available, and computed particle velocities for the group with most of the particles were of the same order of magnitude.

Analysis of the particle trajectories for the different particle diameters showed that the larger particles remain more closely to the symmetry axis, while the smaller particles follow more closely the flow stream lines. The particle velocity is the lowest for the largest particles, and the highest for the smallest particles.

Calculations with radiation showed that the WSSG model gives the most realistic incident radiation. One calculation was made with a particle emissivity, and this let a slight reduced temperatures along the symmetry line. The calculation with radiation also has the tendency to widen the plume, as shown in the temperature profile in the outflow plane. However due to the large flow unsteadiness it is difficult to make a firm conclusion.

4 Conclusions

The GSTP project on the modelling of hot plumes with particles has permitted to extend the NSMB CFD solver with a general equilibrium chemistry module and a particle tracking module, and has permitted to improve and extend the P1-radiation module. In a parallel ESA TRP project the NSMB solver was coupled to the VKI Mutation++ library that permits chemical non-equilibrium calculations for arbitrary mixtures.

The extended NSMB solver was used to make calculations for the VEGA nozzle, for the VEGA launcher and to rebuild hot plume experiments made at DLR Köln.

The calculations for the VEGA Nozzle, VEGA Launcher and for the DLR experiment showed that after the nozzle throat the larger particles remain closer to the symmetry axis than the smaller particles and have lower velocities. Particle velocities on the symmetry axis just downstream of the nozzle exit were measured at DLR Köln, and computed particle velocities were of the same order of magnitude.

The hotplume experiments at DLR Köln showed large unsteady separated flow regions just downstream of the nozzle exit. These large unsteady flow regions makes detailed comparisons difficult.

Calculations with radiation showed that the WSSG model gives the most realistic incident radiation.

Further work is needed to better understand the flow of hot plumes with particles. A more detailed validation study is needed, requiring good quality experimental data, preferably steady, with precise knowledge on the flow conditions including the particle mass flow rate.

References

[1] Zapryagaey, V.I., Lokotko, A.V., Nikiforuv, S.V., Pavlov, A.A., Tchernyshev, A.V., Bannink, W.J., Ottens, H. & Muylaert, J.,
Experimental Investigation of Base Pressure with Hot Supersonic Jet for External Supersonic Flow
Fourth Symposium on Aerothermodynamics for Space Vehicle, 2001, ESA SP-487, 2002.

[2] Murthy, S.N.B., Osborn, J.R.,
Base flow Phenomenon with and without Injection: Experimental Results, Theories and Bibliography
Aerodynamics of Base Combustion, Progress in Astronautics and Aeronautics, V.40: AIAA 1976.

[3] Ludeke, H. and Calvo, J.B.
A Fluid Structure Coupling of the Ariane-5 Nozzle Section during start phase by Detached Eddy Simulation
CEAS Space Journal, Vol 1, 2011.

[4] http://www.esa.int/Our_Activities/Space_Engineering_Technology/Clean_Space

[5] Vos, J.B. and Charbonnier, D.,
TN4100: Validation and assessment of extended CFD solver, March 2019.

[6] Saile, D., Kerkhoff, D., Kühl, V., Steffens, L., Gülhan A., Beversdorff, M., Förster W., Willert, C.
Overview of the ESA-EMAP Project: Characterization of SRM Plumes with Alumina Particulate in Subscale Testing,
International Conference on Flight Vehicles, Aerothermodynamics and Re-entry Missions and Engineering (FAR), 2019.

[7] Vos, J.B. and Charbonnier, D.,
Proposal for Modelling of hot plumes with particles for improved launcher applications,
CFS Engineering Proposal 2015-ESA-GSTP1, June 2015.

[8] http://www.grc.nasa.gov/WWW/CEAWeb/ceaguiDownload-win.htm

[9] Gordon, S. and McBride B.J.
Computer Program for Calculation of Complex Chemical Equilibrium Compositions and Applications I. Analysis.
NASA RP-1311, 1994.

[10] Kee, R.J., Dixon-Lewis, G., Warnatz, J., Coltrin, M.E., Miller, J.A.
A Fortran Computer Code Package for the Evaluation of Gas-Phase, Multicomponent Transport Properties.
SANDIA Report SAND86-8246, 1986.

[11] Scoggins J.B. and Magin T.E.,
Development of Mutation++: Multicomponent Thermodynamic and Transport Properties for Ionized Plasmas written in C++.
AIAA Paper 2014-2966, 11th AIAA/ASME Joint Thermophysics and Heat Transfer Conference, 2014.

[12] Modest, M.F.,
Radiative Heat Transfer, 3rd Edition
Academic Press, 2013.

[13] Hottel, H.C., Sarofim, A.F.
Radiative Transfer
McGraw-Hill, New York, 1967.

[14] Modest, M.F.,
The Weighted-Sum-of-Gray-Gases Model for Arbitrary Solution Methods in Radiative Transfer
Transactions of the ASME, Vol. 113, p.650, 1991.

[15] Atilli, A., Favini, B., Di Giacinto, M.
Numerical Simulation of Multiphase Flows in Solid Rocket Motors
AIAA Paper 2009-5507, 2009.

[16] Kulkarni, A. and Edwards, J.
Development of a Parallel Langrangian Particle Tracking Code for 3-D Multi-Block Curvilinear Grids.
AIAA Paper 2016-0062, 2016.

[17] Troyes, J., Dubois, I., Boric, V., Boischot, A.,
Multi-phase reactive numerical simulations of a solid rocket motor exhaust jet.
AIAA Paper 2006-4114, 2006.